Enhanced Passive Thermal Propulsion System

Dwight Warner
Tusaire Incorporated
11110 Industrial Circle (Suite C)
Elk River, MN 55330
Phone: (763) 633-9177 X12 Fax: (763) 633-9180 email: dwight.warner@tusaire.com

Ellis Warner
Tusaire Incorporated
11110 Industrial Circle (Suite C)
Elk River, MN 55330
Phone: (763) 633-9177 X11 Fax: (763) 633-9180 email: ellis.warner@tusaire.com

Contract Number: N00014-08-C-0455
http://www.tusaire.com

LONG-TERM GOALS

The long term goal is to advance our understanding of thermal energy extraction from the ocean thermocline using an enhanced passive thermal propulsion system. Integration of this new propulsion technology in a low drag hydrodynamic shape is expected to yield undersea glider speeds in excess of 3 Knots (5 Knots may be achievable), and persistence measured in years.

Higher speeds will allow:

- Enhanced glider operations in currents
- Increased measurement rate of ocean parameters of interest
- Reduced timelines to assess parameters of interest in a fixed ocean region

Longer Persistence will provide:

- Potential for underwater glider prepositioning with the ability to "loiter" while waiting for missions
- Reduced logistics manning to deploy and recover vehicles
- Increased measurement area per vehicle deployment

Future technology extensions of this effort will provide propulsion capability in weaker ocean thermocline regions. Hybrid vehicle designs consisting of passive energy extraction and on-board stored energy will likely result, which will increase overall mission flexibility and performance in all oceans.

Report Documentation Page

Form Approved
OMB No. 0704-0188

1. REPORT DATE	2. REPORT TYPE	3. DATES COVERED
30 SEP 2008	**Annual**	**00-00-2008 to 00-00-2008**

4. TITLE AND SUBTITLE
Enhanced Passive Thermal Propulsion System

5a. CONTRACT NUMBER
5b. GRANT NUMBER
5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S)

5d. PROJECT NUMBER
5e. TASK NUMBER
5f. WORK UNIT NUMBER

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
Tusaire Incorporated,11110 Industrial Circle (Suite C),Elk River,MN,55330

8. PERFORMING ORGANIZATION REPORT NUMBER

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)

10. SPONSOR/MONITOR'S ACRONYM(S)

11. SPONSOR/MONITOR'S REPORT NUMBER(S)

12. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution unlimited

13. SUPPLEMENTARY NOTES
code 1 only

14. ABSTRACT
The long term goal is to advance our understanding of thermal energy extraction from the ocean thermocline using an enhanced passive thermal propulsion system. Integration of this new propulsion technology in a low drag hydrodynamic shape is expected to yield undersea glider speeds in excess of 3 Knots (5 Knots may be achievable), and persistence measured in years.

15. SUBJECT TERMS

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT	b. ABSTRACT	c. THIS PAGE			
unclassified	**unclassified**	**unclassified**	**Same as Report (SAR)**	**23**	

OBJECTIVES

The comprehensive objective of this effort is to develop an underwater vehicle that consists of an enhanced thermal propulsion system encased in a low drag hull shape that is capable of harnessing ocean thermal energy to propel the vehicle to speeds approaching 3 Knots.

The specific objectives are:

- Research and Development of an enhanced thermal propulsion system including:
    - Development of an engine including variable ballast
    - Sizing of heat exchangers
    - Investigations pertaining to engine cycle time reduction

- Low drag hull shape:
    - Leverage Navy and DARPA high speed low drag vehicle design methodologies (TAPS Reference 1) and apply them to the current vehicle hydrodynamic design.
    - Utilize State-of-the-Art CFD tools (FLUENT Reference 2) to support and extend the TAPS methodologies to develop a robust design that will meet the drag requirements associated with 3 Knot speed performance.

This activity will leverage previous passive ocean thermal extraction technologies, perform additional research related to thermal extraction, investigate heat transfer related time constants, generate vehicle hydrodynamic designs, build the integrated vehicle with variable ballast propulsion and demonstrate speeds approaching 3 Knots. The major subsystems of this demonstration vehicle (named Kraken) are shown in Figure 1.

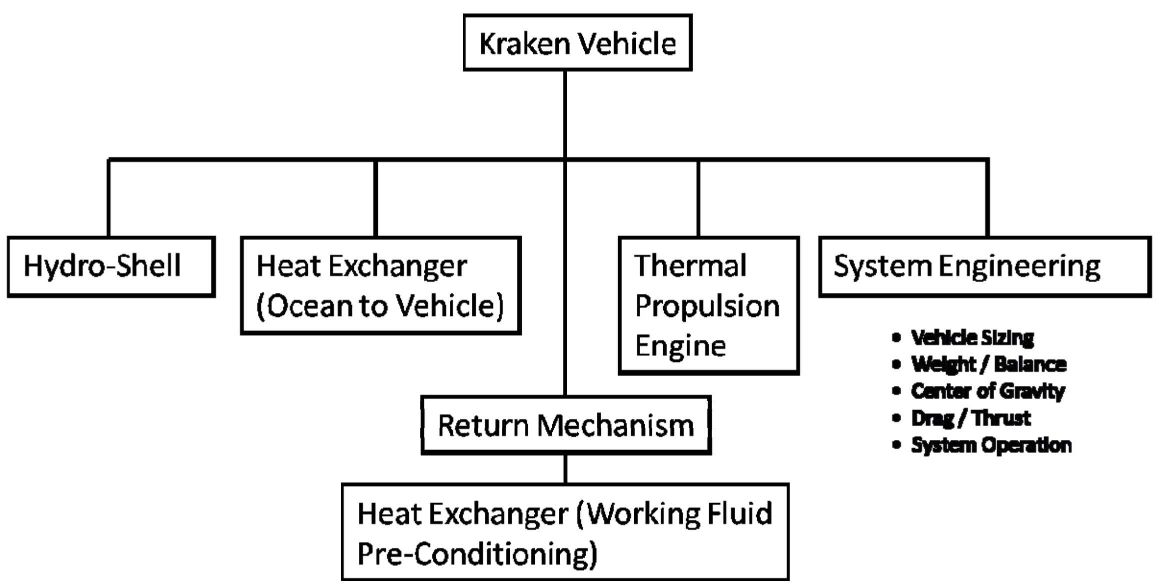


Figure 1. Integrating Key Subsystems using System Engineering, to develop a balanced system, will provide Kraken with a 3 Knot speed capability.

The expected operations are shown in Figure 2, along with the primary and secondary research areas identified with the respective operational domains. As shown, the primary research pertains to the propulsion engine heat extraction from the ocean, time constants related to heat extraction and resetting the propulsion cycle. Secondary research pertains to effective low drag hull shapes that will provide drag at or below propulsion thrust levels.

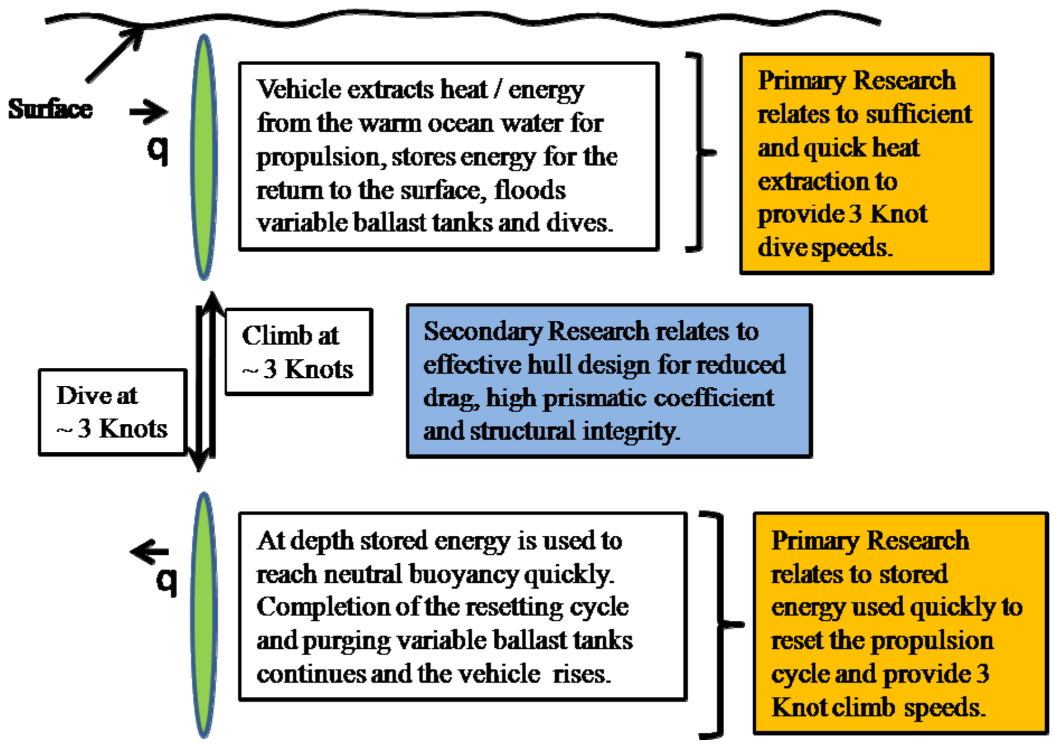


Figure 2. Key Technology Research and Development will lead to 3 Knot Speeds.

APPROACH

Tusaire Incorporated is leading this current research and development technology effort related to the enhanced passive thermal propulsion system. The major subsystems (Figure 1.) consist of the vehicle hydro shell, which is shaped to satisfy the drag requirement and provide structural integrity over the operational depth regime; primary heat exchanger for extracting energy from the high temperature portion of the thermocline; a return mechanism that resets the propulsion cycle; a thermal propulsion engine; and system engineering, to insure a balanced design. A customized System Engineering Trade Study Model (SETSM) is used to assist in developing a balanced set of requirements for the enhanced thermal propulsion and vehicle hull subsystems. The SETSM approach will be discussed first, followed by enhanced thermal propulsion and effective conventional or low drag hull shapes design.

SETSM Approach:

Using SETSM, a balanced set of vehicle requirements and design parameters are generated such that the speed goal is achievable. Once SETSM establishes the requirements and design parameters, then preliminary and detailed design for propulsion, hydrodynamics, and hull structure subsystems are performed. Our SETSM approach is depicted in Figure 3.

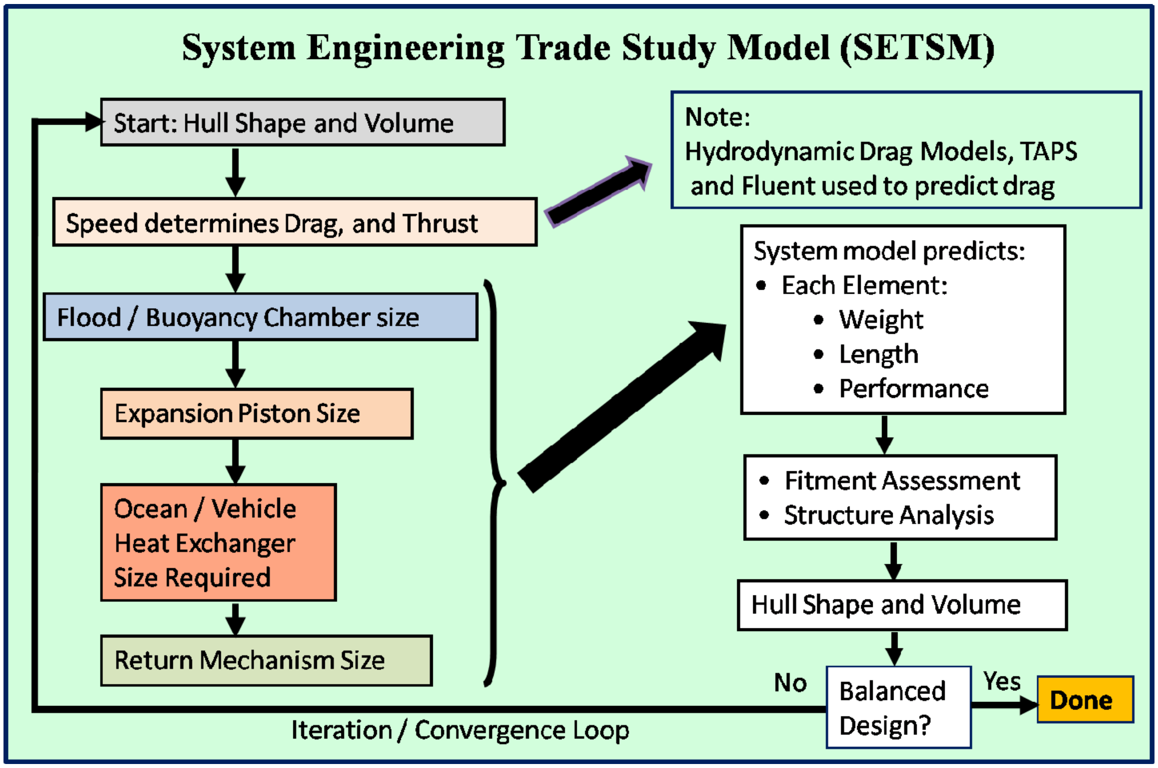


Figure 3. Effective System Engineering Trades can reduce the development time.

SETSM is used to define the design space, minimize weight, fitment problems, and potential re-design efforts. Some design and testing will be used to validate or adjust the SETSM. Using this approach, we are able to look at the sensitivities and interactions of:

- Vehicle shape
- Vehicle predicted drag
- Heat energy extracted from the ocean
- Engine design based on the requirement that engine thrust equals vehicle drag
- Engine support equipment sizing

Enhanced thermal propulsion approach:

Given the System Engineering (SE) flow down requirements (starting requirements) from SETSM, the detailed engine research and development effort is initiated. Our approach consists of the following activities:

- Generate engine sizing and propulsion models to be incorporated into the SE trade study model
- Develop heat transfer model for the ocean / vehicle interaction

- Design and Size the Variable Ballast (VB) flood chamber to provide required thrust for dives and climbs
- Design and Size the heat engine
- Design and Size the Return Mechanism (RM) that resets the engine for the next cycle
- Investigate impacts of non-isothermal conditions associated with the engine
- Assess time constant magnitude associated with VB chamber flooding and storing energy into the RM
- Assess time constant magnitude associated with VB chamber purging and engine resetting
- Determine engine sensitivities to non-isothermal compression and expansion of the working fluids, and perform necessary R&D to minimize any negative non-isothermal conditions
- Design, develop and build a viable thermal propulsion engine system that will provide sufficient thrust to attain 3 Knots and reset the engine cycle

Effective conventional or low drag hull shapes design approach:

Given the System Engineering (SE) flow down requirements[1] (starting requirements) from SETSM, the detailed Hydrodynamic design effort is initiated.

The approach used for Hydrodynamic design (for this current effort) consists primarily of vehicle drag prediction. Drag for low speed incompressible flow is segregated into three categories for computational assessments:

- Viscous Flow (Surface Skin Friction)
    - Laminar (reduced drag)
    - Turbulent (higher drag)
- Boundary-Layer Transition (location that marks the change from laminar flow to turbulent flow)
- Pressure Drag (Including flow separation if present)

We use a synergistic / checking methodology to develop the hydrodynamic vehicle design. This approach is based primarily on the use of the TAPS[2] code and the FLUENT CFD code. This synergistic / checking methodology is shown in Figure 4. Geometry generation is performed using Tusaire's geometry codes.

---

[1] These initial requirements for Hydrodynamics, are vehicle size, speed, and allowable drag levels.

[2] TAPS was developed by McDonnell Douglas under Navy funding in the 1970's. It was the work horse code for developing the high speed laminar flow vehicles during this time frame. It is an excellent tool for prediction of laminar flow, boundary-layer transition location, turbulent flow, and the associated drag. However the TAPS code is limited to axisymmetric shapes, and does not predict flow / drag affects associated with flow separation, and does not predict drag associated with angle-of-attack flows. Where TAPS is weak FLUENT is strong. FLUENT handles 3-d geometries, angle-of-attack flow fields, and predicts flow separation and the associated pressure drag quite well. FLUENT is basically considered the State-of-the-Art fluid dynamic CFD code, however we consider TAPS more effective in predicting laminar flows and boundary-layer transition locations.

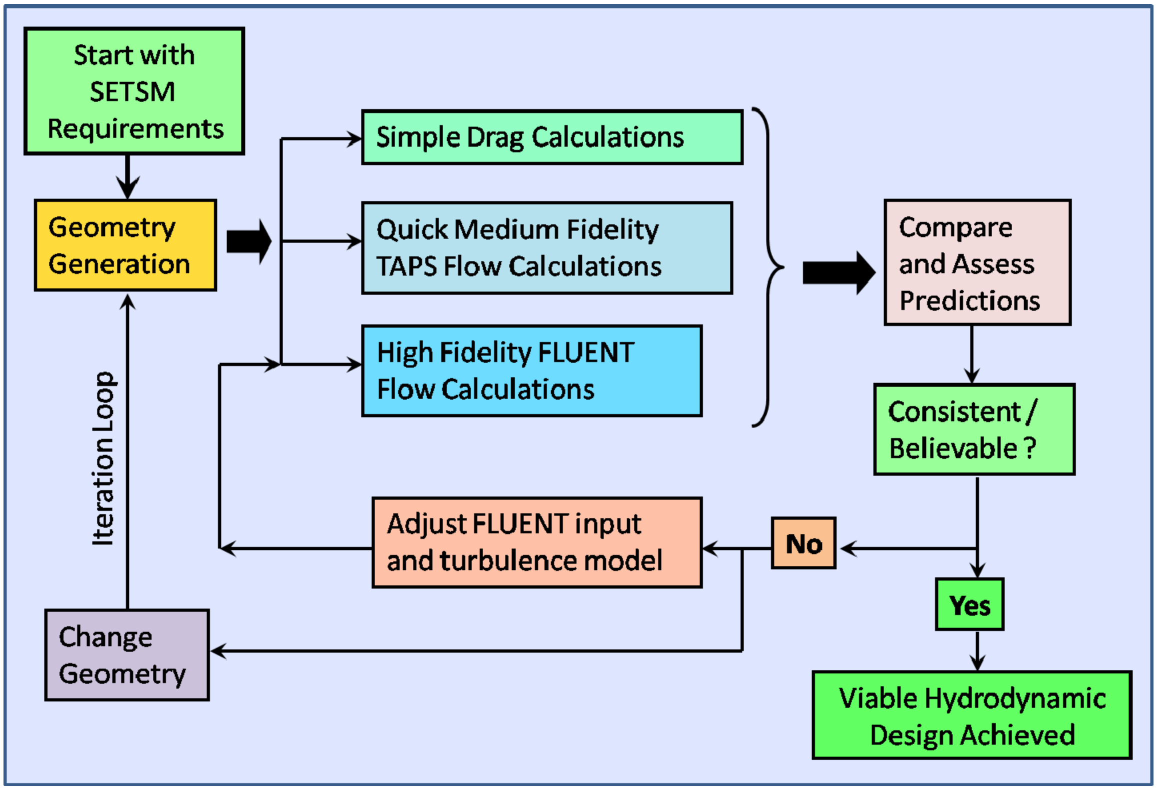


Figure 4. Effective hull shape design is obtained through our Synergistic / Checking approach associated with simple, medium, and high fidelity CFD Codes.

WORK COMPLETED

The work completed is summarized in these three categories:

1. System Engineering and sizing
2. Thermal Propulsion
3. Vehicle Hydrodynamics, Hull Structure and Fitment

System Engineering and Sizing:

The SE method, discussed in the approach section, is used to guide the vehicle design and development. Output from this SETSM is shown in Figure 5, for a 22.86 cm (9 inch) diameter vehicle that is 3.048 m (10 feet) long. Every point represents a design modeled from the parameter variations within SETSM. The red symbols track displacement to weight ratio for the parameter variations investigated. A value > 1 is a viable design, meaning ballast mass must to be added to bring the vehicle to neutral buoyance (therefore there is weight margin). The green symbols track length available to length required (obtained from model parameters). Values > 1 are viable (hardware is predicted to fit, or there is length margin). The current baseline vehicle is predicted to provide a weight margin > 15%, and a length fitment margin of > ~ 5%.

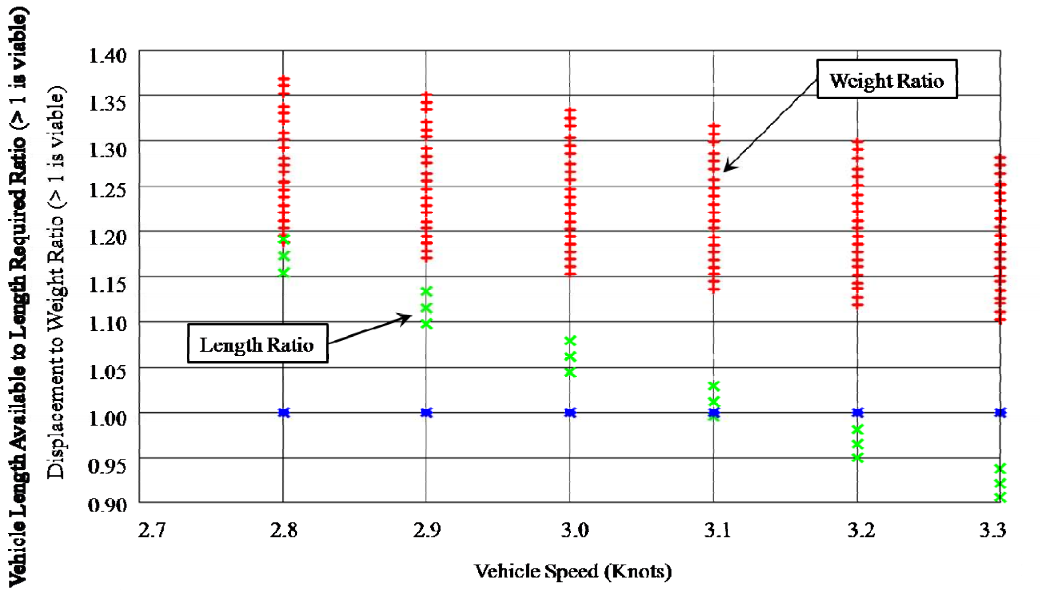


Figure 5. Our baseline configuration is predicted to be viable for 3 Knot vehicle speeds.

Current baseline vehicle resulting from SETSM:

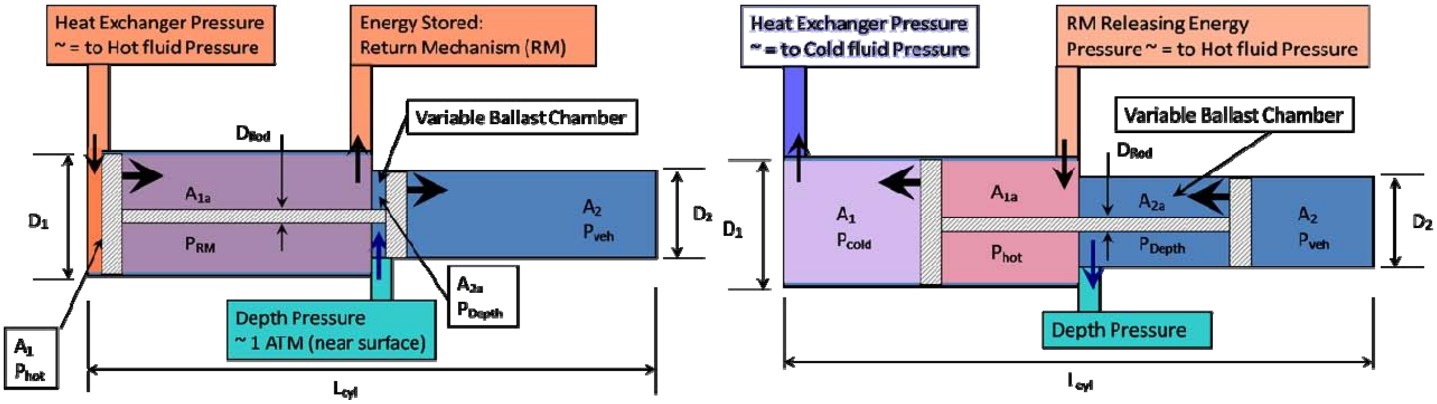
The current baseline from SETSM for a 3 Knot vehicle with a prismatic coefficient of ~ 0.883 is provided in Table 1.

Table 1. Key baseline requirements and design parameters

	Weight	Length	Diameter
Vehicle	~ 113.6 kg (250 lbs)	~ 3.048 m (10 ft)	~ 22.9 cm (9 in)
Return Mechanism	~ 3.2 kg (7.1 lbs)	~ 62.2 cm (24.5 in)	~ 16.73 cm (6.6 in)
VB displacement	~ 2.1 kg (4.6 lbs)	~76.2 cm (30 in)	~ 5.7 cm (2.25 in)
Heat Exchanger Coil	~ 10.5 kg (23 lbs)	~ 53.3 cm (21 in)	~ 1.9 cm (0.75 in)
Engine	~ 14.5 kg (32 lbs)	~76.2 cm (30 in)	~ 15.24 cm (6 in)

Thermal Propulsion:

The engine concept for surface and deep operations is depicted in Figure 6. On the surface heat is transferred from the ocean to the working fluid (WF) in the coil, which drives the engine piston from left to right, this stores pressure-volume (PV) energy into a second WF contained in the return mechanism (RM), and floods the VB chamber with water providing downward thrust. At depth PV energy is released from the RM, driving the engine piston from right to left, resetting the engine and purging the VB chamber water. This cycle provides upward thrust.



a) Engine Cycle on the Surface **b) Engine Cycle at Depth**

Figure 6. Sketch of Engine Operation at the Surface and at Depth.

Time Constants associated with the Thermal Cycle:

Tusaire is developing technologies for the rapid extraction (~2.8 kJ/sec, 2-3 min)[3] of thermal energy from the ocean, as well as rapid storage and release of that extracted energy over a similar time scale. The thermal engine is composed of a Working Fluid (WF), a Highly Compressible Fluid (HCF), an Exterior Coil Heat Exchanger (ECHE), and a Return Mechanism (RM). The ECHE contains the WF that expands when heated by warm oceanic surface water, resulting in a pressure increase. When the ECHE is cooled, the fluid contracts with a resulting pressure decrease. The ECHE is responsible for extraction of ocean energy. The extracted thermal energy is converted to Pressure-Volume (PV) energy which is stored in the RM via the (HCF).

The current HCF has been chosen for property characteristics of a homogeneous single phase, with high compressibility during near constant pressure compression, and a relatively high critical pressure. Tusaire developed a Redlich-Kwong cubic equation of state code (following Reference 3 and 4 methods) for selecting fluid candidates. Each candidate is then assessed, using NIST REFPROP (Reference 5), for quantitative fluid properties and beneficial use in mixtures of fluids. We tailor the HCF to operate with expected ocean thermocline temperatures near equatorial oceans (the Tropics).

The return mechanism stores PV work at the surface and releases that work at depth. The rate of storage and release of energy from the return mechanism has presented some technical challenges related to adiabatic heating on compression and cooling on expansion. The Pressure density plot (Figure7) compares initial experimental compression with NIST isotherms and indicates that the HCF under compression does not behave in an isothermal fashion. It was learned in Laboratory experiments that the HCF was relatively sensitive to small changes in temperature (on the order of 3-4°Celsius). The adiabatic heating and cooling term of compression and expansion changed the temperature by more than 3° Celsius, resulting in non-beneficial pressure variation in the return mechanism, which increased the time constant associated with the cycle. These results led to the development of a Return Mechanism Research Activity to investigate non-isothermal behavior. Several configurations of the RM were conceived for testing and are shown in figure 8.

[3] It is a desired goal to reduce the time constant as much as possible.

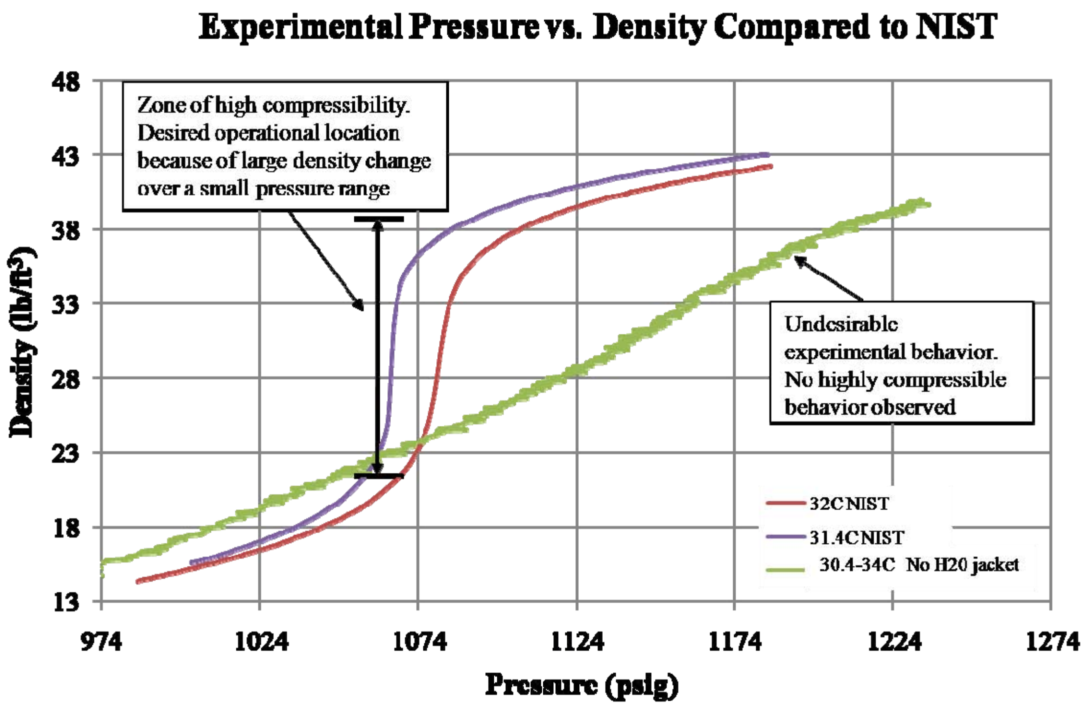


Figure 7. Note experiment had poor agreement with NIST model. The experimental compression did not have a region of high compressibility (large density change over a small pressure change), which made it unsuitable for the return mechanism.

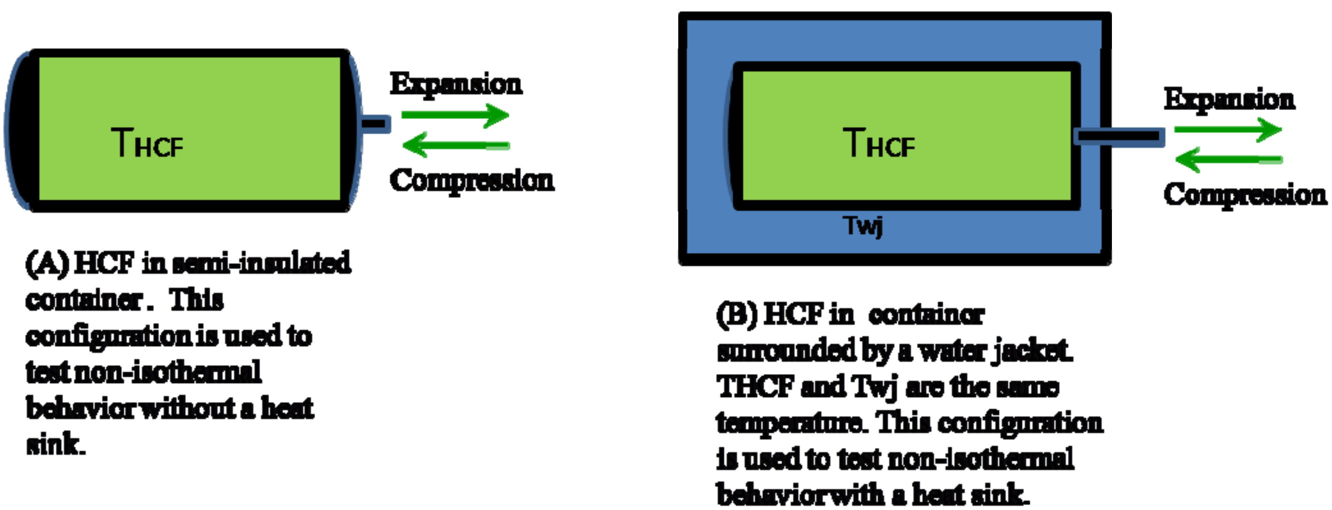


Figure 8. Experimental hardware thermodynamic configurations used for determining the effect of non-isothermal compression and expansion on pressure. In addition, these configurations are used to support additional research and development efforts.

The Return Mechanism Research Activity (figure 9) shows the methods used to diagnose and modify the thermodynamics of the return mechanism. Previous efforts have made the assumption that adiabatic heating on compression would have a negligible effect on the pressure density behavior of the HCF in the return mechanism. This assumption was proven to be inaccurate over shorter cycle times[4]. The two experiments to date, configurations (A) and (B), show that there is a significant difference in thermal and isobaric properties between the two experiments for the same time interval of compression and expansion. The water jacket resulted in significant improvements in the isobaric zone of high compressibility and was in relatively close agreement with NIST (figure 10).

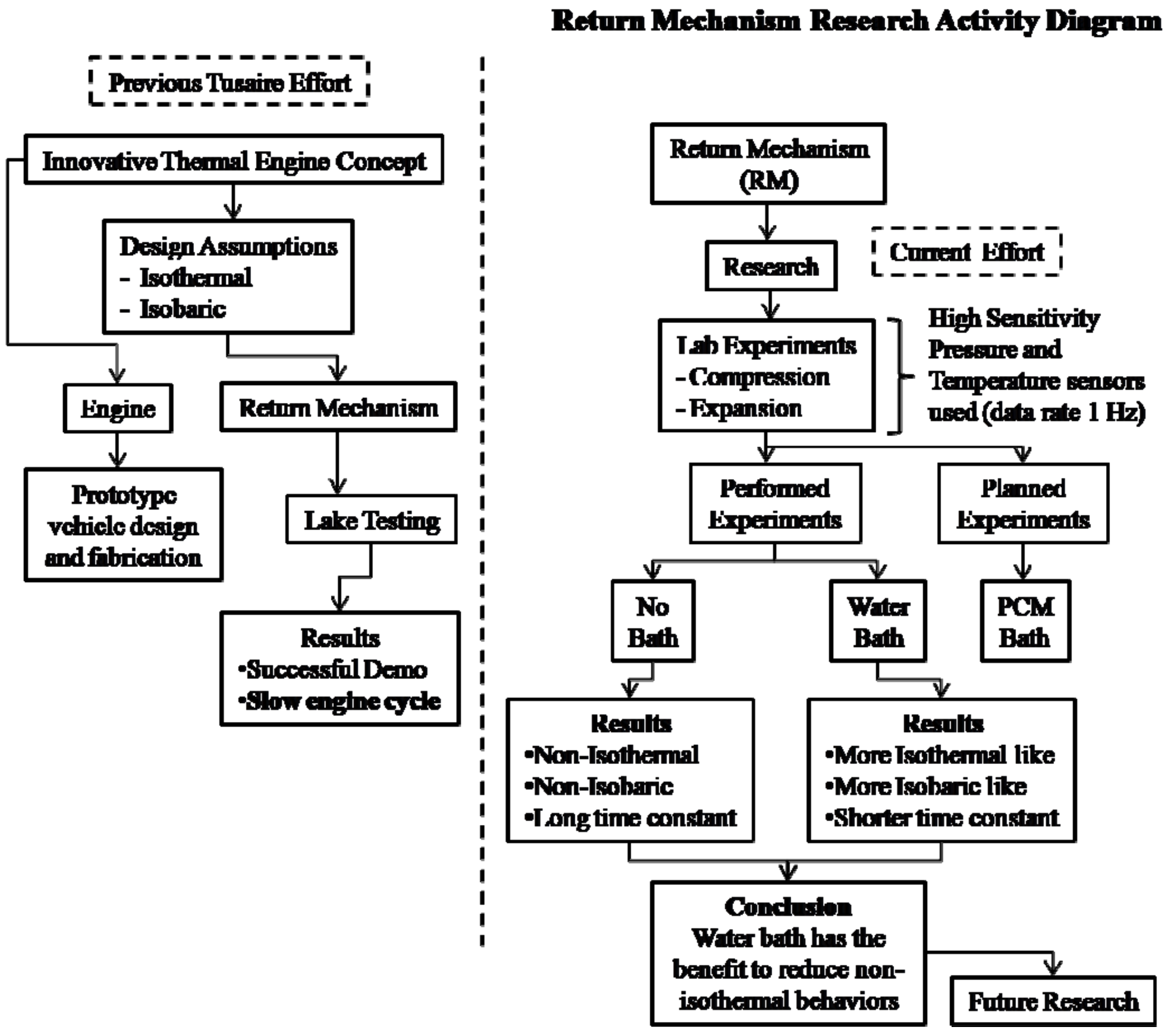


Figure 9. Activity diagram shows Research that led to improved understanding of non-isothermal (real world) compression and expansion of thermodynamic working fluids.

[4] Tusaire laboratory testing using the configuration (A) setup discovered non-isothermal and non-isobaric behavior in the HCF which resulted in increased cycle time constants.

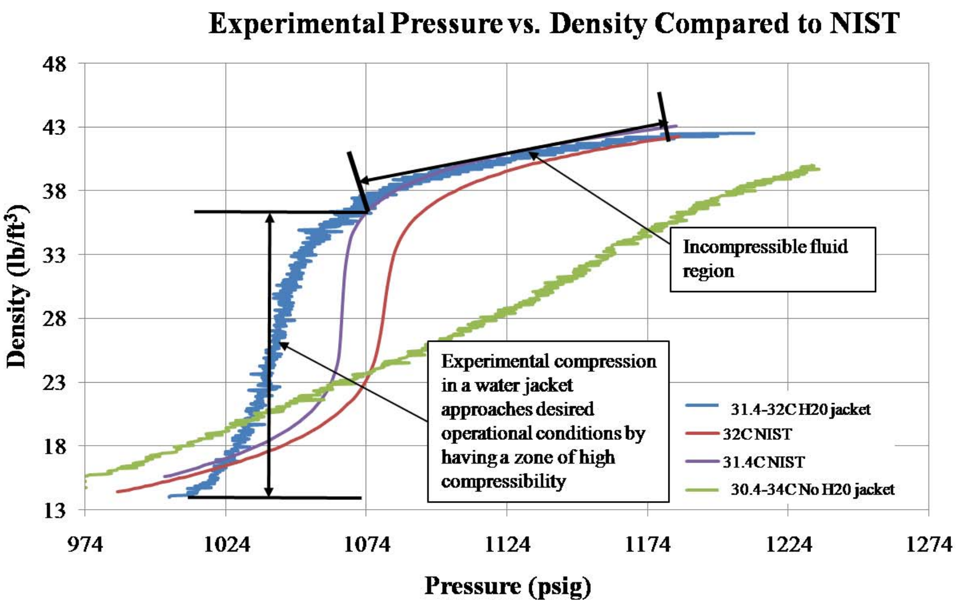


Figure 10. Note how the experimental water jacket (blue) curve has a zone of high compressibility and is in relatively close agreement with NIST.

More experiments with isothermal water jackets have been performed. Although these resulted in improved pressure density curves, some challenging trends still exist as a result of adiabatic heating and cooling. Figures 11 and 12 show results related to compression and its associated impact on temperature and pressure. Investigations of rate affects during expansion, and the associated impact on temperature and pressure are shown in figures 13 and 14.

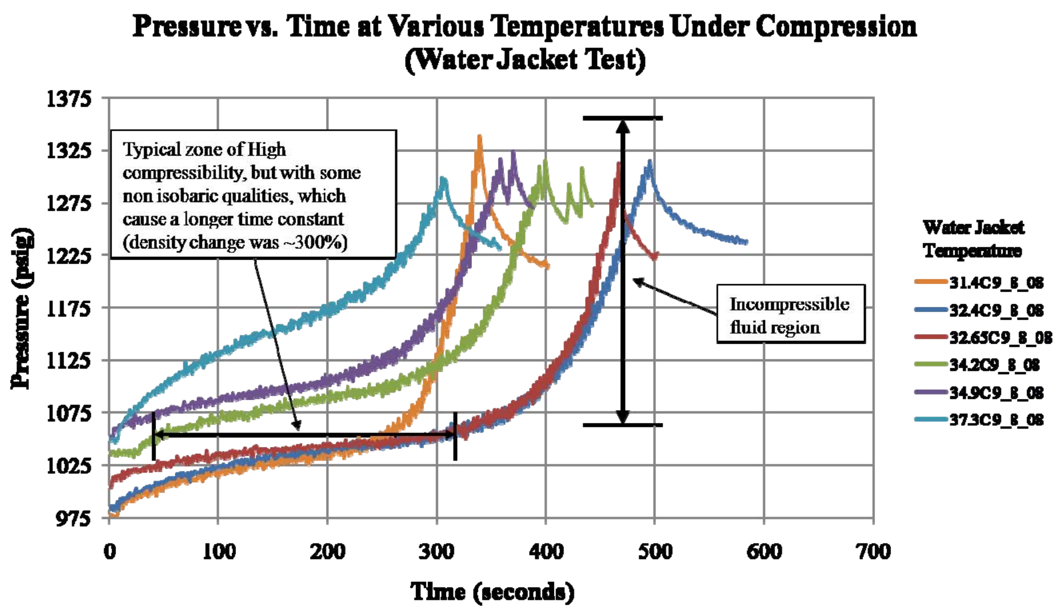


Figure 11. The pressure increased by approximately 50 psi, over 350 seconds of compression, which was approximately 4 times better than the data with no water jacket; however, it is still less compressible than NIST predicts. Higher compressibility makes the fluid more suitable for the Return Mechanism.

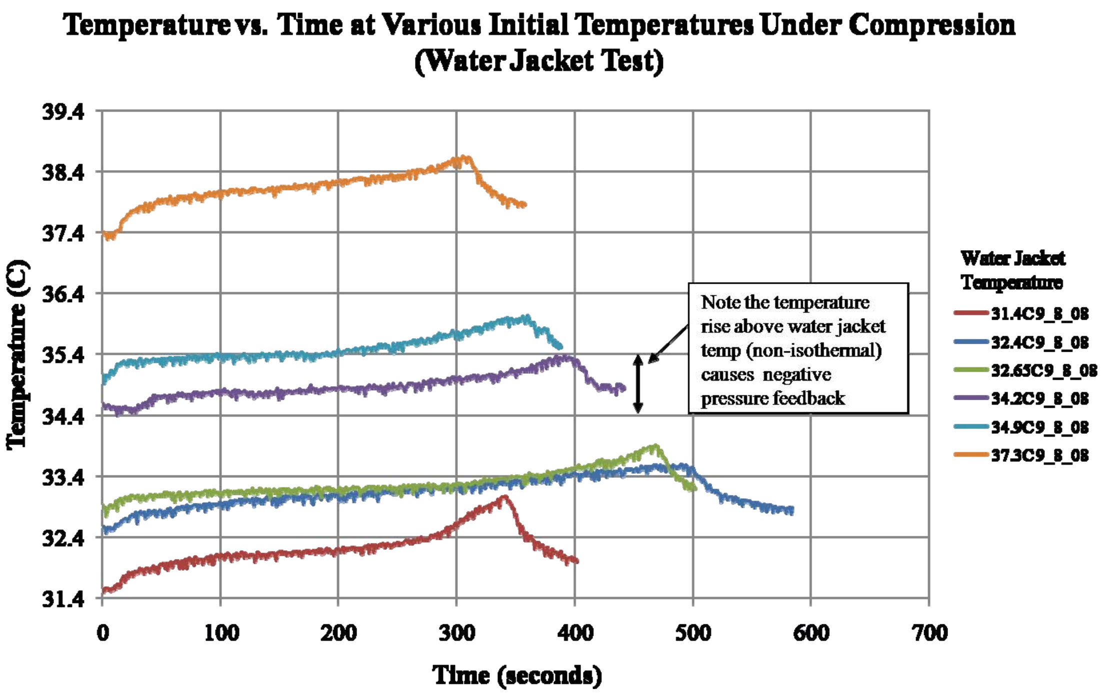


Figure 12. Even in an isothermal water jacket, the temperature of the HCF rises about 1 Celsius above the water jacket temperature. This temperature rise occurs over 400 seconds of compression. The adiabatic temperature rise resulted in a pressure rise that increased the cycle time constant.

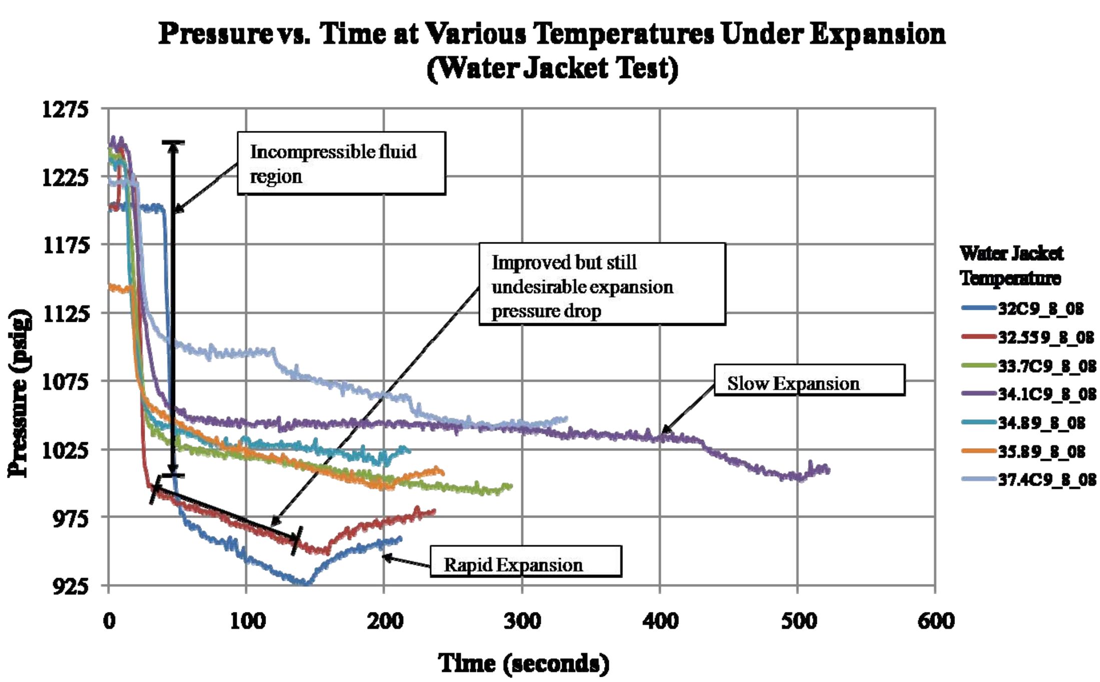


Figure 13. Note the time dependence of pressure with expansion. The faster the HCF expands, the more prominent the pressure drops. Pressure drops will increase the cycle time constant.

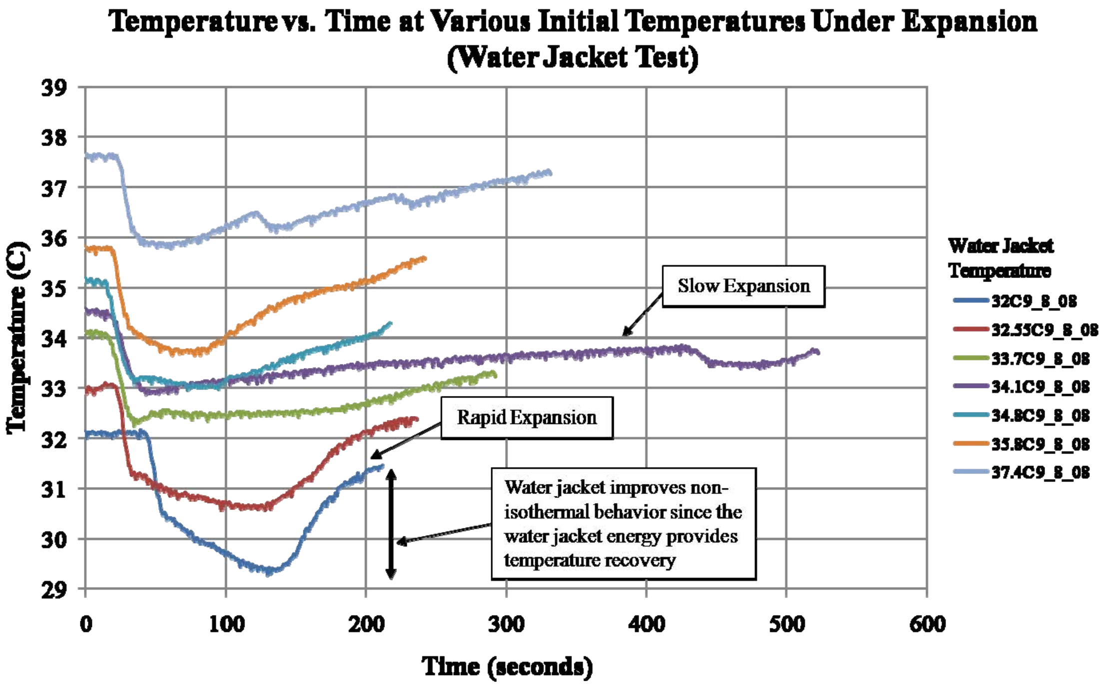


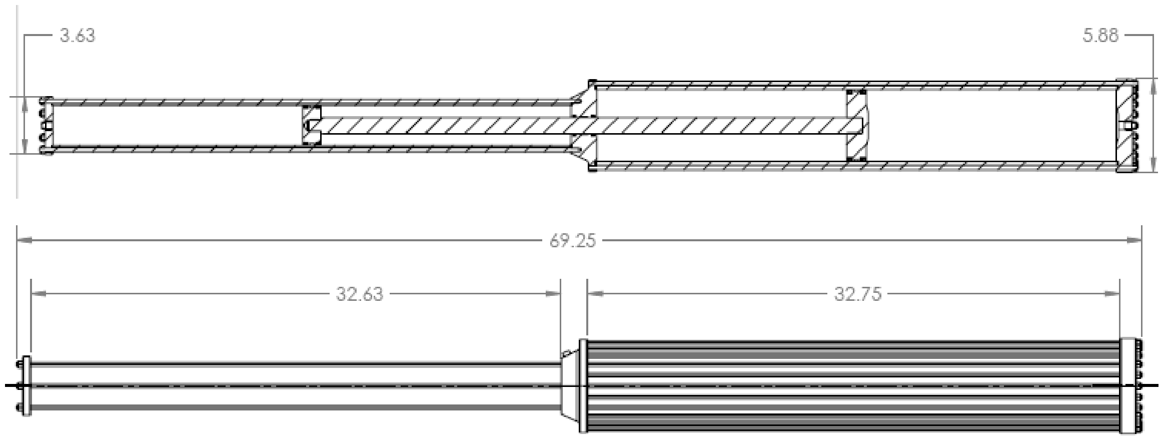
Figure 14. Note the time dependence of temperature with expansion. The faster the expansion, the more prominent the temperature drop is in the HCF. Also, notice the temperature recovery with time as the water jacket transfers heat to compensate.

Testing with water jackets has shown that the initial negative pressure behavior can be reduced by approximately four times over a non-jacketed return mechanism. Additional research experiments and testing will be required to assist return mechanism hardware concepts and designs for the vehicle that will provide the desired time constant.

Engine with VB Chamber Design:

We have developed an engine and VB chamber design based on the suggested requirements from SETSM.

The "built to requirements" for the engine consist of a primary bore of 4.5 inches, flood chamber bore of 2.5 inches, operational pressures of 1200 and 900 psi respectively, and a total weight < 33 pounds. The design is shown in Figure 15 (a), and the fabricated engine with VB chamber is shown in Figure 15 (b). The total length is ~ 6 feet, and consists of carbon fiber composite with self lubricating piston seals and the tie rods consist of Al 7075-T6. This engine can produce thrust levels of $\pm$ 10.23 N ($\pm$ 2.3 pounds).



(a)

(b)

Figure 15. An Engine with VB chamber has been designed and developed (a) and built (b) that provides the required thrust for 3 Knots operations.

Heat Exchanger Research and Development:

Most steel or aluminum tubing that is available for heat exchanger coils is produced in nominal 20 ft lengths. If a heat exchanger requires greater than 20 ft of tubing, a high pressure leak proof joint needs to be produced. Tusaire's research determined that a welded joint was extremely difficult and expensive to produce because of thermal stress on the metal and X-rays of the welded joints were necessary to guarantee the weld. We investigated alternative methods for building long heat exchanger coils, and developed an approach which works, does not require X-ray inspection, and has no negative material property degradation. Our method is based on the use of adhesives instead of welding. Both sides of the coil to be joined are preformed with small injection ports installed for the injection of the adhesive. This results in a leak proof, high pressure joint that has been tested to 3500 psi. Our heat exchangers will operate at pressures below 1500 psi, well below the levels that have been demonstrated. One of our fabricated coils is shown in figure 16 below.

Figure 16. Advanced Coil Fabrication using Loctite H8000™

Hydrodynamics:

Following the method shown in the Approach Section for hydrodynamics, numerous computational runs were performed with trial geometry shapes, providing the pressure distribution and drag predictions from TAPS. When a result looked promising, we modeled it in detail and performed additional computations using FLUENT, running on an HPC computer system. A typical prediction of the pressure coefficient from TAPS and FLUENT is shown in Figure 17, and shows very good agreement. The main difference is in the afterbody region, where the boundary layer is thick and minor flow separation occurs[5].

[5] The TAPS potential flow does not account for this thickened boundary layer impact on Cp; however, FLUENT predicts a slight pressure field difference in the afterbody region, which is expected and validates the synergistic use of these two codes.

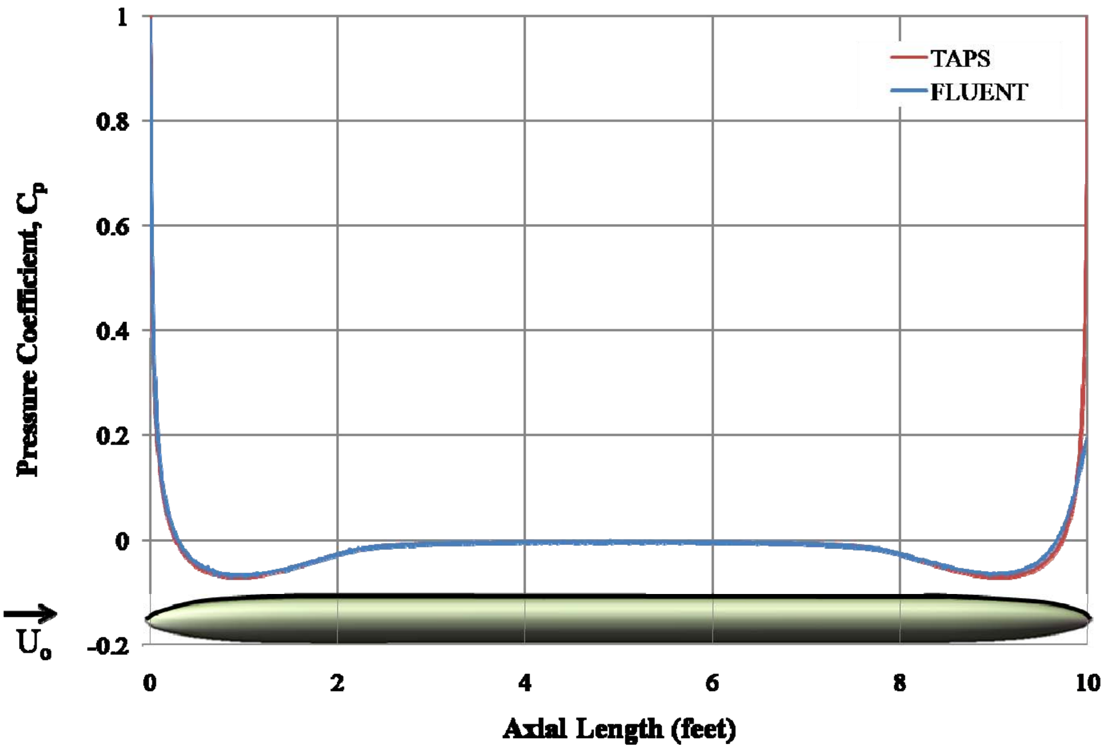


Figure 17. Excellent agreement of the vehicle pressure coefficient has been obtained from TAPS and FLUENT. As expected, FLUENT with its coupled *Navier-Stokes* ***solver is able to predict the influence of the thick boundary layer on the Cp near the end of the vehicle, where the boundary layer is thickest.***

Hydrodynamic analyses have been performed for the environment and fluid properties shown in Table 2.

Table 2. TAPS and FLUENT Environmental / Operational Parameters.

Medium	Salt Water
Angle of attack: α	0° to 10°
Salinity	3.5%
Velocity: U_o	2.96 knots (5 ft/s)
Temperature: T	8.89°C (48°F)
Density: ρ	1027.91 kg/m^3 (64.17 lbs/ft^3)
Dynamic viscosity: μ	0.001435 kg/m-sec (0.0009640 lb/ft-sec)
Additional FLUENT Modeling Parameters	
Model	Fully turbulent, realizable k-epsilon equations
Near wall	Enhanced wall treatment
Grid	Complete 3D, tetrahedral mesh, ≈ 9 million cells

To develop a viable and robust hydrodynamic low drag shape, Tusaire investigated numerous vehicle shapes, their fluid interactions, and resultant drag for various locations of boundary-layer transition, angle-of-attack, and time dependent flow fields. Viewing these computational outputs as a whole allow us to determine the overall robustness[6] of the hydrodynamic design. Our computational outputs for the baseline vehicle are provided below and consist of:

- Pressure Distribution (Figure 18.)
- Drag as a function of boundary-layer transition location (Figure 19.)
- Boundary-Layer Transition location based on the e^9 stability theory to predict laminar flow performance, and the ability to achieve low drag. (Figure 20.)
- Flow field sensitivities to angle-of-attack (Figure 21.)
- Drag sensitivities to angle-of-attack for turbulent (high drag) flow. (Figure 22.)
- Transient flow field predictions, to assess drag sensitivities to "unsteady" flow (Figure 23.)

Our baseline vehicle shape and resultant pressure distribution is shown in Figure 18. The mild adverse pressure gradient (B) is similar to flat plate flow, and at these speeds and unit Reynolds Number, this gradient is not sufficient to prevent laminar flow from reaching the vehicle midbody (Arc-Length Reynolds Number is 2.3×10^6). After the midbody, the favorable pressure gradient (C) is able to extend the laminar flow into the afterbody. Drag predictions as a function of transition location are presented in Figure 19. Drag values range between 8.7 N for all turbulent flow to 2.1 N for laminar flow. These drag values include both friction and pressure drag. Based on the e^9 stability theory, (Reference 6) boundary-layer transition is predicted to occur at 2.85 meters, as shown in Figure 20. Flow field sensitivities to angle-of-attack variations are provided in Figure 21, which show that these angle-of-attacks have a minor impact on flow separation. The drag as a function of angle-of-attack (for full turbulent flow condition – worse case) is presented in Figure 22, and shows that drag increases are less than 10% for angle-of-attacks up to 5 degrees. Time dependent transient flow field calculations (from FLUENT) are presented in Figure 23. These demonstrate that the afterbody flow field changes slightly over time, but the changes are very minor and no significant drag increase was noticed.

Based on these results, we are confident that we have developed a viable low drag vehicle for 3 Knot speeds. Laminar flow is predicted to extend to ~ 93% of the vehicle length. Drag increases associated with angle-of-attack is predicted to be less than 10%. It should be pointed out that laminar flow is not necessary to achieve our speed goal. If laminar flow is achieved (a most likely outcome) then higher speeds above 3 Knots will be demonstrated.

[6] By robustness we mean an assessment of vehicle sensitivities to "off-design" flow conditions, and the resultant drag. For example some vehicles perform very well for "on-design" flow conditions – but may have an order of magnitude drag increase when they operate in an "off-design" state – such as an airfoil in stall.

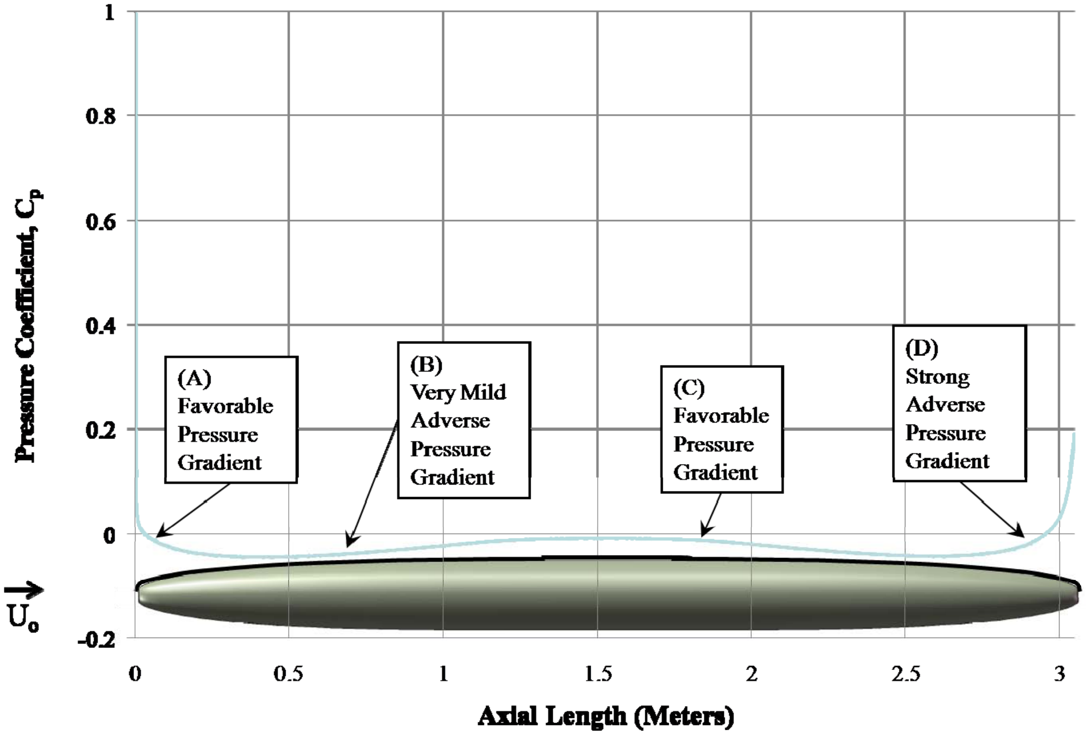


Figure 18. Our selected baseline vehicle shape leads to high prismatic coefficient, and an excellent pressure distribution that has the potential to provide low drag.

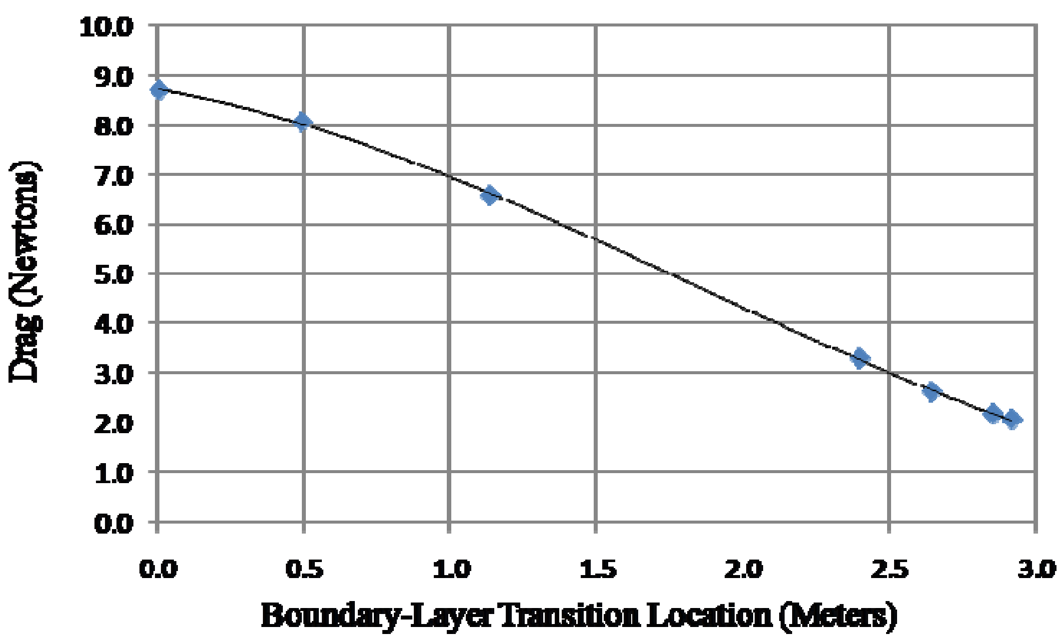


Figure 19. Vehicle drag is substantially reduced when boundary-layer transition is delayed.

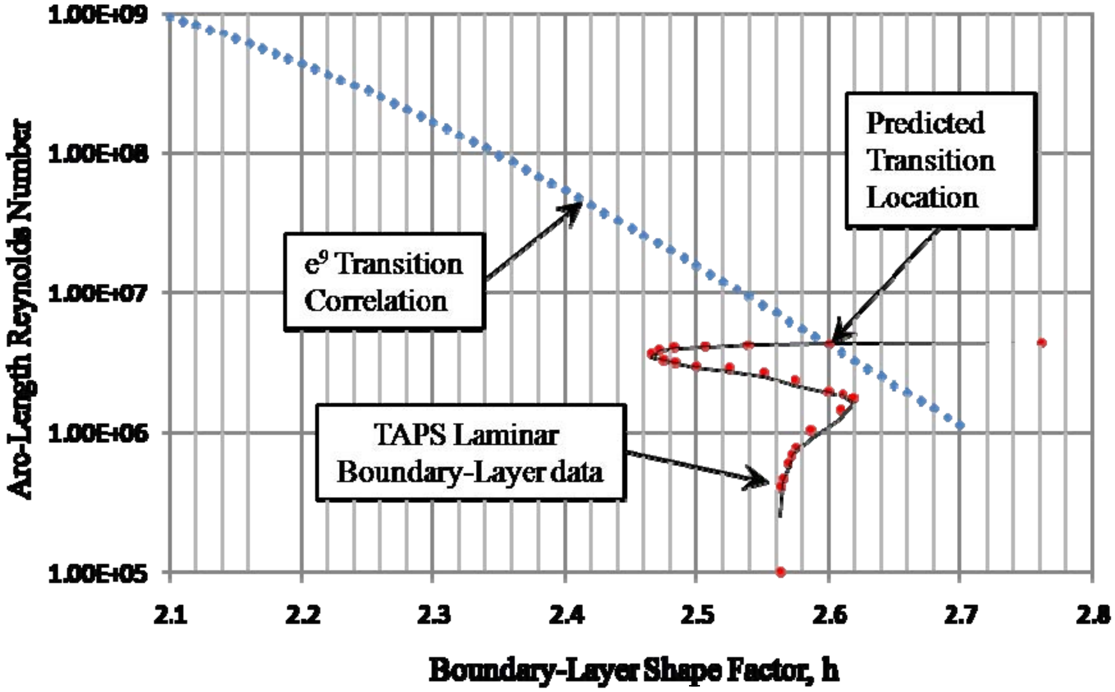


Figure 20. Laminar flow Transition is predicted to occur at h = 2.6. This corresponds to an axial location of 93% of total length, and results in a drag prediction of 2.1 N.

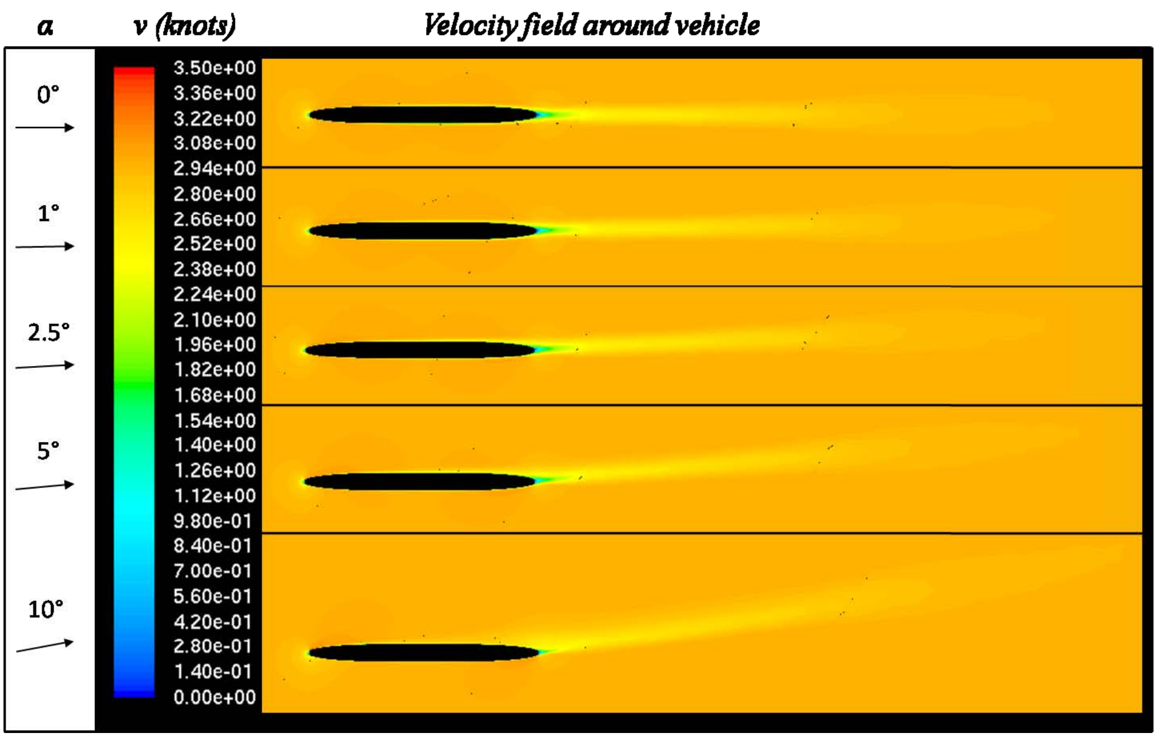


Figure 21. Velocity field contours around the vehicle for varying angles-of-attack, α. As angle-of-attack increases, the trailing edge wake moves to the leeward side and vehicle drag increases slightly.

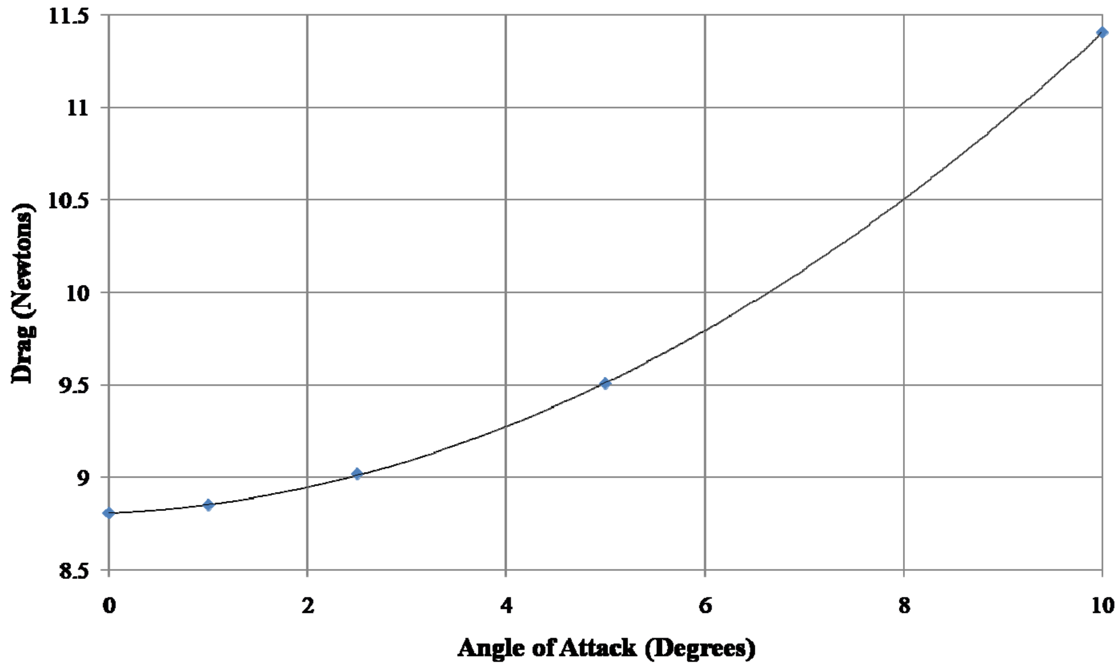


Figure 22. Drag variations as a function of Angle-of-Attack, α, for an all turbulent flow case. The Drag increase is less than 10% for α less than 5 degrees. These results bound the drag increase associated with pressure drag increases from α variations. Our vehicle system is design to handle these levels of pressure drag increases.

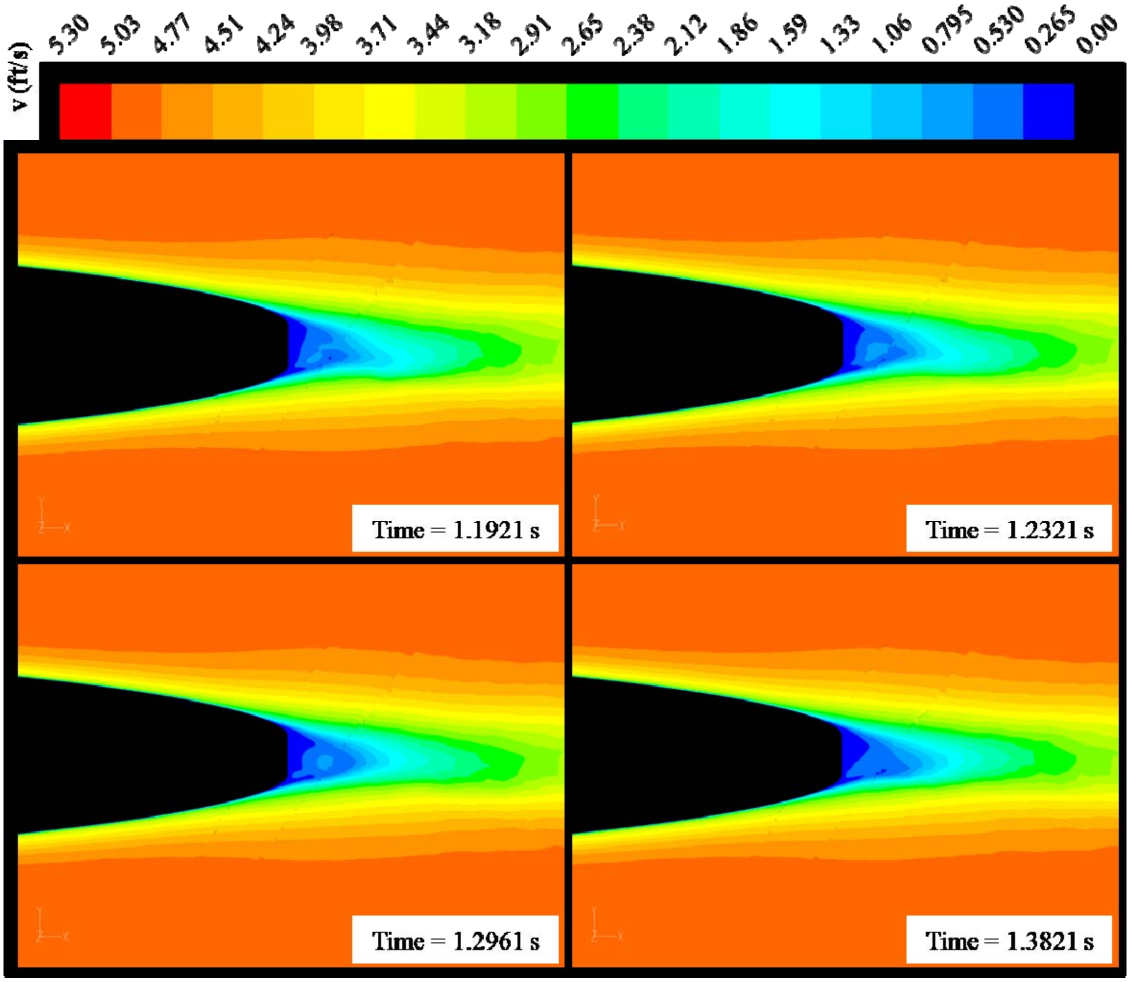


Figure 23. FLUENT transient flow calculations show that the afterbody flow is very stable, and drag fluctuations were not observed at zero angle-of-attack. This demonstrates that our vehicle shape selection has very stable drag at a given speed. We do not expect off-design flow field conditions to cause any major drag increases.

Vehicle Fitment Design:

Vehicle fitment involves the management of space within Kraken. Solid Works® (Reference 7) has been used to perform detailed fitment, weight, and center of gravity determination. Figure 24, shows the current fitment of all components within the low drag hull shape. All components of the baseline system fit in the current hull design and require installed ballast to achieve neutral buoyancy.

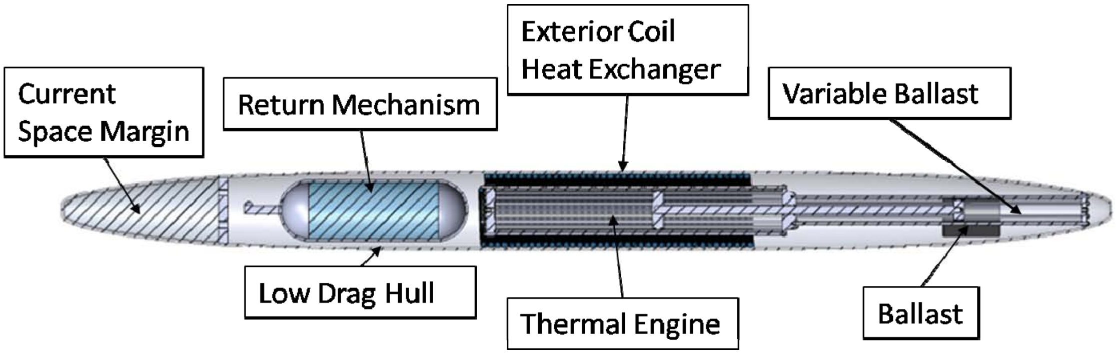


Figure 24. Fitment diagram, from Solid Works®, showing that the system components fit within the hull shell.

RESULTS

Our meaningful technical results achieved in 2008 are:

- Results to date indicate that the goal of attaining 3 Knot speed performance from propulsion based on energy extraction from the thermocline is achievable. This is based on a combination of system engineering analysis using heat engine and thermodynamic laboratory test data, and hydrodynamic predictions using state-of-the-art tools and methodologies.
- We have learned that non-isothermal behavior (temperature changes less than 5^o C) can have negative impacts on cycle time constants. However, a path forward has been identified and research started that has the potential to remove these negative impacts.
- Developing a balanced design using SETSM reduces time to reach viable vehicle size determination that satisfies all requirements, and reduce any potential redesign efforts.
- Confidence in the final hydrodynamic shape and associated drag is achieved through our design approach using TAPS and FLUENT hydrodynamic codes in a synergistic / checking method.
- High pressure heat exchanger coils can be fabricated from the use of room temperature adhesives, reducing time, and reducing the cost associated with conventional welded heat exchangers.

IMPACT/APPLICATIONS

Potential future impact for Science and/or Systems Applications is the substantial increase in underwater glider speeds and persistence. Current thermal gliders operate in speeds measured in a few tenths of Knots. This enhanced thermal propulsion technology has the potential to provide speeds approaching 5 Knots in the future.

REFERENCES

1. The Transitional Analysis Program System (TAPS-Code), Arvel Gentry, Douglas Aircraft, 1976

2. FLUENT release 6.3.26 copyright 2006, Fluent Inc.

3. Physical Chemistry A Molecular Approach, Donald A. Mcquarrie, John D. Simon, University Science books, 1997

4. The Properties Of Gases & Liquids 4th, Robert C. Reid, John M. Prausnitz, Bruce E. Poling, Mcgraw-Hill, 1986

5. NIST standard reference database 23, version 8.0 E.W. Lemmon, M.L. Huber, M.O. Mclinden Physical And Properties Division 2007

6. "Laminarization of Bodies of Revolution Moving in Water," A.M.O. Smith, A.R. Wazzen, W. C. Hsu, school of engineering and applied science University of California, 1977

7. Solid Works® office premium 2008 sp4.0 copyright 1995-2008 Dassault Systèmes